

10/13/92


SAFSIM OVERVIEW

by Dean Dobranich

**Sandia National Laboratories
Nuclear Technology Department**

October 1992

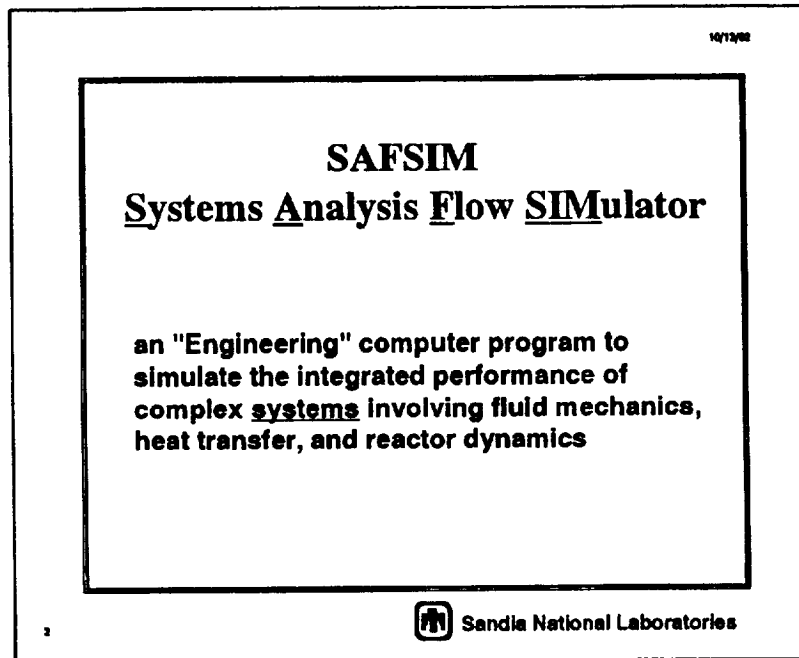
1

 Sandia National Laboratories

An overview of the SAFSIM computer program is provided in this presentation.

SAFSIM is being developed at Sandia National Laboratories and is currently funded by the Air Force SNTP program.

Slide 1



SAFSIM is a general purpose, FORTRAN computer program to simulate the integrated performance of complex systems involving fluid mechanics, heat transfer, and reactor dynamics. SAFSIM provides sufficient versatility to allow the engineering simulation of almost any system, from a backyard sprinkler system to a clustered nuclear reactor propulsion system. SAFSIM is based on a 1-D finite element model and provides the analyst with approximate solutions to complex problems.


Although SAFSIM can be used to model specific components in detail, its major strength is the ability to couple multiple components together to investigate synergistic effects between components. This is important because, in general, a system of optimized components does not produce an optimum system. Non-linearities in the physics can produce system performance that might not be expected from analysis of an isolated component.

Slide 2

10/13/92

Desired Program Attributes:

- ✓ **versatile**
- ✓ **fast running**
- ✓ **robust**
- ✓ **quality assessed**
- ✓ **documented**
- ✓ **benchmarked (when possible)**
- ✓ **transportable (FORTRAN77)**

 Sandia National Laboratories

SAFSIM is being developed with versatility as its primary attribute. Thus, it can be used to assess the performance of a variety of user-defined systems on a consistent and unbiased basis.

Speed and robustness are also key attributes that are incorporated in the overall development goals of SAFSIM.

SAFSIM documentation, benchmarking, and quality assessment are ongoing activities.


SAFSIM has been run on a VAX8650, a Sun Spark station, and an HP9000 workstation in addition to a 486/25 PC on which it is being developed.

Slide 3

10/13/92

Basic Physics Modules

- Fluid Mechanics
- Structure Heat Transfer
- Reactor Dynamics

 **Sandia National Laboratories**


Three basic physics modules are included in the current version of SAFSIM: (1) Fluid Mechanics (solution of the conservation equations governing single-phase fluid flow), (2) Structure Heat Transfer (solution of the heat conduction equation for solid structures), and (3) Reactor Dynamics (solution of the time-dependent equations governing nuclear reactor neutron density, including reactivity feedback and decay heat). These three physics modules are described more fully in the following charts.

Slide 4

10/13/92

Fluid Mechanics

- **1-D Finite Element Model**
 - compressible thermal energy equation with advection/conduction/convection
 - compressible mechanical energy equation
- **Multiple, user-specified, liquid or gas flow networks**
- **Single phase with ideal gas, polynomial, or user-supplied equation of state options**
- **Multiple gases with mixing models**

 Sandia National Laboratories

The fluid mechanics physics module is based on a 1-D finite element model and solves the conservation of mass, momentum, and energy equations for a single-phase fluid. Compressible or incompressible fluids can be simulated. Thermal and mechanical energy equations are solved iteratively to provide the solution to a total energy equation.

The 1-D finite elements can be connected in series or parallel to create any desired flow network. Multiple networks can be included to model, for example, a heat exchanger with gas on one side and liquid on the other.

The user can select the equation of state for the different fluids in all networks. Choices are: ideal gas, polynomial function of temperature (for incompressible fluids), and user-supplied. An interface is in place within SAFSIM to facilitate inclusion of a user-specified equation of state. Thus, an understanding of the internals of SAFSIM is not required to add an equation of state.

Mixing models are provided to allow simulation of multiple gases in a network. Thus, different gases can be tracked throughout a network and fluid properties for the mixture are automatically determined.

Slide 5

Fluid Mechanics (continued)

- Porous media finite element
- Compressor/Pump element
- Special choked flow boundary element
- Distributed flow manifold element (with options for transpiration flow and tees)
- Super element capability
- Automatic K-factors for expansions and contractions
- Open or closed networks

 Sandia National Laboratories

Special finite elements allow simulation of flow in porous media, compressors/pumps, and manifolds. Also, a special element allows implementation of a choked flow boundary to model a nozzle. The manifold element includes options to automatically account for transpiration flow (blowing/sucking conditions) and branching flows with respect to friction factors and heat transfer coefficients.

Super elements allow a series of finite elements to be combined into one "super element". This greatly increases computational speed for solution of the mechanical energy equation. Accuracy is also improved because a smaller matrix is produced, resulting in less round-off error.

K-factors are automatically determined for expansions and contractions if desired. Separate K-factors can be included for both forward and reverse flow for each finite element. Also, additional l/d can be added to account for bends, obstructions, etc... A relative wall roughness can also be included.

Both open and closed networks can be modeled.

Slide 6

Fluid Mechanics (continued)

- Convection based on log-mean delta-T
- Upwinding with automatic determination of upwind factors based on Peclet number
- Pressure, mass flow rate, temperature , zero heat flux, and mass fraction boundary conditions
- Three matrix solvers
 - Gauss-Seidel, iterative
 - Cholesky decomposition, direct
 - Gauss elimination, direct



Sandia National Laboratories

Convection heat transfer in the thermal energy equation is based on the log-mean temperature difference which increases accuracy, especially for low flow simulations. To accomplish this, a special technique was developed to allow the linear, 2-noded elements of SAFSIM to provide the accuracy of a higher order element with minimal extra computational expense.

Upwind elements are used for solution of the thermal energy equation. The optimum upwind factor is determined for each element based on the Peclet number, which provides a measure of advective dominance. Thus, problems that are advectively or conductively dominated can be simulated.

Boundary conditions for the fluid mechanics solution can be specified at any node in the network.

Three numerical solvers are provided to add robustness. The user can select a solver or let SAFSIM execute the three solvers in succession until a solution is achieved.

Slide 7

Structure Heat Transfer

- 1-D Finite Element Model
 - automatic timestep control
 - subtimesteps for each structure
- Automatic spherical, cylindrical, or rectangular geometry finite element generator via input if desired
- Temperature-dependent properties
- Automatic implicitness factors



Sandia National Laboratories

The structure heat transfer module is based on a 1-D finite element model and solves the heat conduction equation for solid structures (pipe walls, plates, fuel rods or particles, thermocouples,...). Automatic timestep control can be selected for each structure if desired and each structure can have its own subtimestep. Thus, structures with large time constants can run at large timesteps and are not forced to run at the small timesteps required of structures with much smaller time constants.

Although geometry input must be completely specified by the analyst, automatic mesh generation is provided for structures with spherical, cylindrical, or rectangular geometry.

Conductivity and specific heat can be temperature dependent if desired and several options are available for specifying property values, including tables, polynomials, and power laws.

The implicitness factor is automatically determined for all nodes of each structure, at each subtimestep. This ensures that the best accuracy is achieved for any given timestep.

Slide 8

Heat Transfer (continued)

- Multiple exchange surfaces for each structure finite element
- Extensive built-in HTC correlation library
 - laminar and turbulent flows
 - internal and external flow geometries
 - gases, liquids, and liquid metals
- Temperature, heat flux, and convective/radiative boundary conditions



Sandia National Laboratories

Each finite element can have multiple exchange surfaces. An exchange surface allows heat transfer between the structure and the coolant (via convection or radiation) or between different structures (via radiation or conduction). For example, a structure finite element representing a pipe wall may have one exchange surface to model forced convection heat transfer to a coolant flowing through the inside of the pipe and another exchange surface to model free convection to another coolant on the outside of the pipe. A third exchange surface could be added to model radiation to the outside coolant, if desired.

SAFSIM allows the analyst to select a HTC correlation for laminar flow conditions and another for turbulent flow conditions for each exchange surface. A built-in library contains over 90 correlations including internal and external flow geometries. Correlations for gases, liquids, and liquid metals are included. Also, an interface is provided to allow the analyst to easily add her own correlations.


Either temperature, heat flux, or convective/radiative boundary conditions can be used for each structure.

Slide 9

10/13/92

Reactor Dynamics

- Point (0-D) Kinetics Model with feedback
 - multiple reactors
 - adaptive timesteps
- Multiple feedback coefficients for fuel, moderator, control rods/drums ...
- User-specified precursor and decay heat groups (automatic concentration initialization if desired for steady state)
- Euler or fifth-order Runge-Kutta solvers

 Sandia National Laboratories

The reactor dynamics physics module is based on a point (0-D) kinetics model and includes reactivity feedback and decay heat. Multiple reactors can be specified and multiple feedback coefficients are allowed for each reactor to account for all system interactions. The analyst has complete control over how the feedback coefficients are defined. Multiple reactors can be coupled via user defined feedback coefficients if desired. Also, special-purpose "control laws" can be added to the program to simulate reactor startup and shutdown transients. Adaptive timestep control can be employed. A source term also can be included.


Any number of delayed neutron groups and decay heat groups can be specified. Initial precursor concentrations can be input or calculated automatically by SAFSIM based on steady-state conditions.

Two solvers are available for integration of the reactor dynamics equations: (1) Euler, and (2) Runge-Kutta-Fehlberg (RKF). The analyst can switch between solvers during a problem if desired.

10/13/92

Miscellaneous

- Automatic steady-state option
- Function-controlled variables
- User-supplied subroutine interfaces:
 - functions
 - equation of state and fluid properties
 - heat transfer coefficients
 - reactor dynamics control laws
 - special-purpose input and output

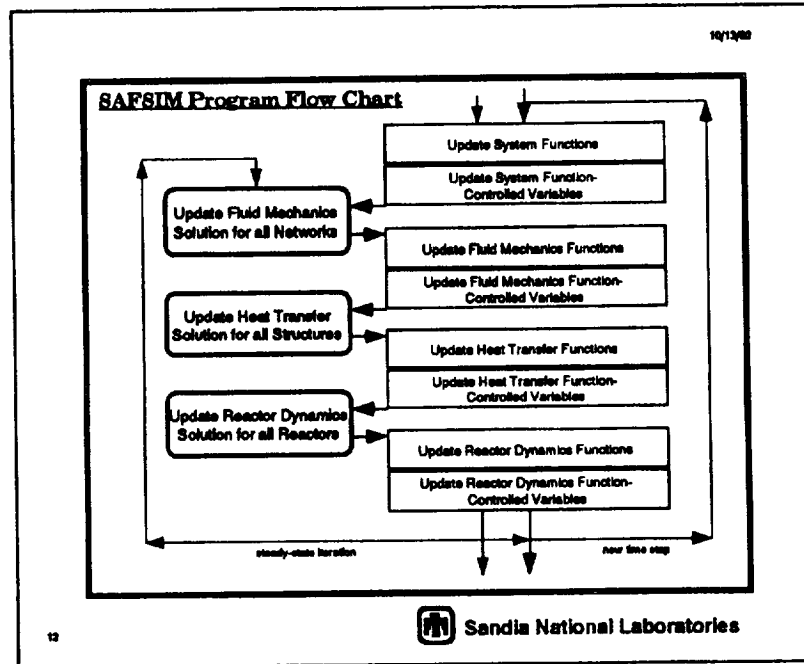
 Sandia National Laboratories

Although SAFSIM is a time-dependent computer program, it can be used to perform steady-state calculations. Two methods are available. The first method is to simply run a transient simulation until the time derivative terms are sufficiently small. SAFSIM offers a second method in which the time-derivative terms are set to zero and wall temperature iterations are performed to obtain consistency between the fluid mechanics and structure heat transfer physics modules. This automatic steady-state method can be combined with the first method if desired.

Function-controlled variables are a unique feature of SAFSIM that allow the analyst to specify most of SAFSIM's input variables as functions of any of its output variables. An extensive library of mathematical functions is available within SAFSIM or the analyst can add his own. For example, flow lengths and areas can be specified as functions of structure temperature to simulate expansion effects.

SAFSIM provides 5 user-supplied subroutine interfaces to allow the analyst to tailor SAFSIM to problem-specific modeling needs. These interfaces streamline the process of adding special subroutines.

Slide 11




This chart provides a top level flow diagram of SAFSIM and indicates the computational sequence for both steady-state and transient analyses. The three physics modules, along with function-controlled variables and functions, are explicitly coupled to simulate the integrated performance of an entire system. Employing explicit coupling between the different physics modules (which all may have vastly different characteristic time constants) greatly increases program versatility. For very rapid transients the system timestep can be decreased to more tightly couple the different parts of the system.

10/13/92

Program Status

- All physics modules operational
- Cleanup and enhancements in progress
- Benchmarking and documentation in progress

 Sandia National Laboratories

SAFSIM is a functioning computer program and is currently being used to solve a variety of problems at Sandia National Laboratories. However, SAFSIM is not complete and additional development is anticipated. Benchmarking and documentation are extremely big tasks that are expected to proceed concurrently with development.

Three manuals are planned to document SAFSIM: (1) a theory manual that will contain a description of the governing equations and numerics; (2) an input manual that contains a complete description of all of the input variables required to build an input model; and (3) an application manual that will provide benchmark problems in addition to several example problems. The input manual (Sandia National Laboratories internal report SAND92-0694) is complete and is being distributed as of October, 1992.

Slide 13

Future Enhancements

- Turbine element
- Built-in bandwidth minimizer for mechanical and thermal energy equations
- Blowdown tank option
- Structural Mechanics Physics Module
- LU decomposition with iterative refinement for large networks
- Restart capability



Sandia National Laboratories

To expand the class of problems for which SAFSIM is applicable, several enhancements are planned:

- addition of a turbine finite element
- a built-in bandwidth minimizer to increase the speed and accuracy of execution
- a boundary condition option to allow easy and quick simulation of tank blowdown
- a structural mechanics physics module based on a 1-D finite element model to predict the linear and nonlinear stress-strain behavior of solid structures, including plasticity and creep
- addition of an LU decomposition solver with iterative refinement to account for roundoff error when modeling extremely large networks
- restart capability to allow continuation of a problem

Slide 14

Future Enhancements (continued)

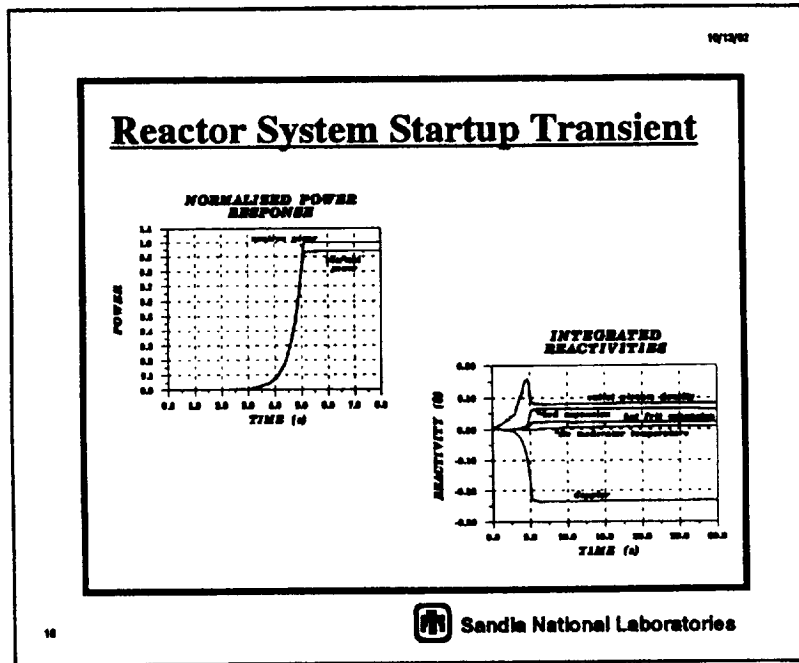
- Kaganove solver for reactor dynamics
- 2-D tables and other special functions
- Pre- and post-processing (graphical)
- Dynamic temperature, mass flow rate, and density terms in fluid mechanics equations
- Upwind elements for the mechanical energy equation
- Liquid metal modeling options



Sandia National Laboratories

- addition of a Kaganove solver for long-duration reactor dynamics problems
- 2-dimensional table capability for functions along with many other special mathematical functions to enhance modeling capability
- graphical pre- and post-processing routines to facilitate input model building and output interpretation
- addition of all the dynamic terms in the fluid mechanics module
- addition of upwind elements to the mechanical energy equation to allow simulation of supersonic flow
- input options to allow simulations involving liquid metals (such as an accumulator and an electromagnetic pump)

Slide 15




These graphs show results of a SAFSIM application in which a system based on a particle bed reactor is brought to full power in 5 s. In addition to the particle bed fuel element, the moderator, reflectors, vessel, and control drums are modeled. The MIT-SNL control law is used to control the startup of the reactor. Feedback effects due to coolant density, fuel temperature, moderator temperature, bed and hot frit expansion, and control drum rotation are included in the model. The input model includes 64 fluid mechanics finite elements, 145 structure heat transfer finite elements, and 1 nuclear reactor. The problem was run on a 486/25 MH PC and required 4 minutes of CPU time to simulate 30 s of transient time. The average timestep was about 5 ms for the fluid mechanics. The same problem required 30 s of CPU time on an HP9000 workstation.

10/13/92

Current Applications of SAFSIM

- SNL
 - PBR System Startup/Shutdown Transients
 - PBR Element Performance
 - NET Simulation
 - ETS Simulation
- NASA
 - Simulation of NERVA NRX/EST System

 Sandia National Laboratories


This chart (and the next) lists several applications of SAFSIM that are in progress and demonstrates the versatility of SAFSIM. Simulation of the NERVA NRX/EST system is the only application so far that has experimental data for an entire propulsion system for comparison to SAFSIM calculation. The model is being built at NASA/Lewis and currently contains 240 fluid mechanics finite elements. Agreement with experimental data is excellent.

10/13/92

Current Applications (continued)

- B&W
 - PBR Element Performance
 - Reactor System Performance
- Grumman
 - Propulsion System Control Studies

18

 Sandia National Laboratories

SAFSIM applications in progress. (see preceding chart)